Creative Commons License

ZERØ TO HERO™

A New User's Guide to MakerBot's

Replicator 2

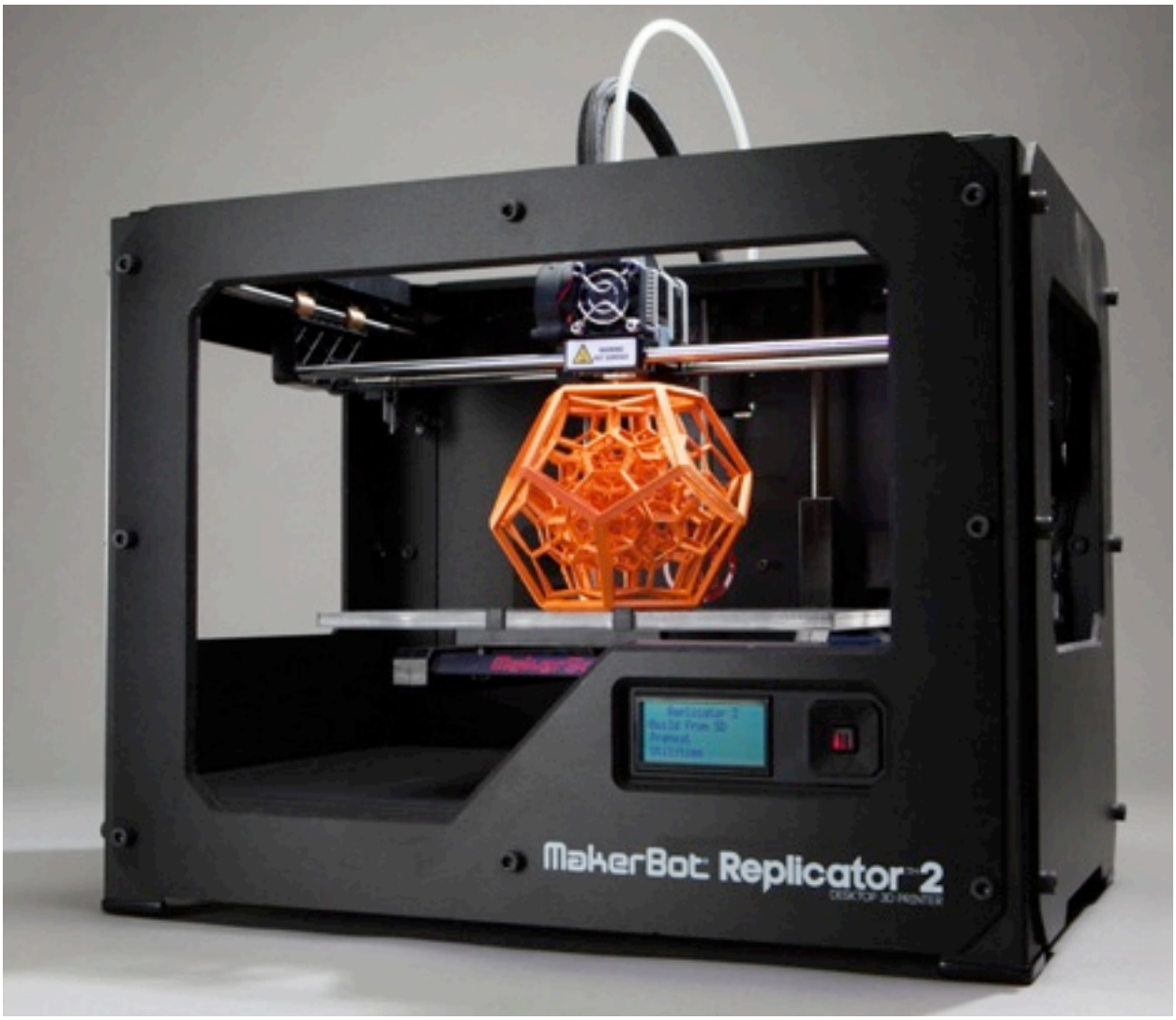


Photo: MakerBot Industries website.

DigiFabLab has no relationship to MakerBot Industries. LLC. *This guide is intended to assist new users who wish to learn more about, and get the most out of, their Replicator 2. Information provided is the result of testing by DigiFabLab and insights from hundreds of Replicator 2 users, shared generously in user forums. While the information provided is believed to be accurate, users assume sole responsibility for employing methods suggested—some of which are at variance with MakerBot's recommendations.*

» QUICKSTART: USEFUL REMINDERS FOR REPLICATOR 2 USERS

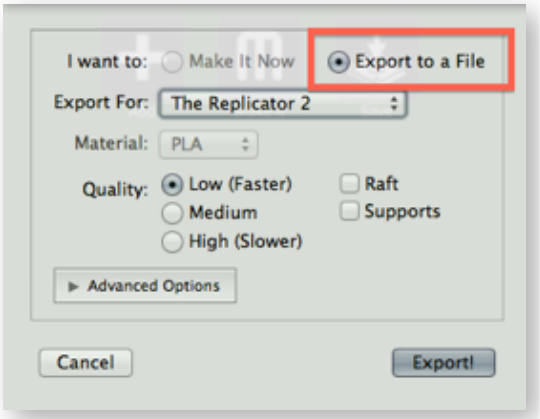
1. Open Object in MakerWare

- Must be in .STL or .OBJ format.
  - Must be watertight, with no gaps.
  - All face normals must be oriented in the same direction (out is best).
- 'Add' Button » {Select desired file}

2. Verify Size, Scale, and Location

- Default Units are in mm.
- Scaling from inches to millimeters may be performed in MakerWare.
  - Scale » Inches -> mm
- Make sure object is on the platform
  - Move » On Platform

3. Click 'Make It' Button



- Select Export to a File
- Export For: The Replicator 2
- Select Low, Medium (recommended), or High Quality (not recommended).
- Select Raft if desired.
  - Raft provides a base and better adhesion, but requires more cleanup and will affect the finish.
- Select Supports if needed.
  - Supports are needed if there are any horizontal geometries or elements projecting less than 45° from the build plate.
  - Geometries greater than 45° are self-supporting and do not need supports.
- Advanced Options allows default values to be adjusted.
- Export (to create .X3G toolpaths)

4. Save .Thing file and close MW.

5. Put .X3g file onto SD Card

6. Turn on Replicator2

- Switch located on back

7. Insert SD Card into Printer

- Slot is behind LCD display

8. Wipe Build Plate with Alcohol

9. Verify that Build Plate is Level

- If not, scroll to Utilities » Level Build Plate » Enter

10. Is Filament Loaded?

- If not, scroll to Utilities » Change Filament » Load » Enter

11. Scroll to "Build from SD" » Enter

12. {Select Your File} » Enter

13. Be sure first layers adhere to plate.

- Should lay down a thin, flat bead.

For in depth details, see ZERØ TO HERO sections

» INTRODUCTION:

ABOUT:

The goal of this guide is to help novice users achieve remarkable print results by better understanding 3D printing methods and specific aspects of the Replicator 2.

MakerBot Industries, is among a group of companies that aim to put 3D printing in the hands of every-day users.

The Replicator 2 is a single head printer using PLA (Polylactic Acid) filament—a biodegradable, organic plastic. MakerBot recently introduced an experimental dual head printer—the Replicator 2X—which prints with ABS and possibly other materials.

COMMON NEW USER ISSUES:
FILAMENT FEEDING & PRINT ADHESION

While the Replicator 2 is capable of very high quality printing, it has a few known challenges which resourceful users may easily address. Proper initial setup is crucial for optimal print results. *[See INITIAL SETUP, TROUBLESHOOTING, and MODS sections for more information.]*

NOTE: The OEM extruder mechanism is not addressed in this guide due to its setup challenges. ***DigiFabLab recommends that new users upgrade this immediately.*** Order the upgrade kit from MakerBot (requesting ABS parts). Or tackle your first project and visit Thingiverse.com, download, print, and install the Replicator 2 Extruder Upgrade designed by whpthomas[2], or MakerBot's new derivation of this design, the Replicator 2 Extruder Alternative. *You'll be glad you did!*

CONTEXT:

3D printing began in the late 1970's. Today, with technologies becoming more affordable (and patents expiring), this area is currently experiencing explosive growth—largely due to experimental efforts by the open-source community. DIY kits vary widely and require a degree of skill to assemble and use. RepRap.org, founded in 2005 by Dr. Adrian Bower. is ground-zero for open-source efforts of many types and well-worth checking out.

MakerBot's roots are within the open-source community; however, the Replicator 2 is a significant departure from this tradition. It is not a DIY kit, but a pre-assembled, proprietary consumer product with its own simplistic slicing software — MakeWare.

MakerBot has been celebrated (by *Wired, BBC, Gizmodo, Make,* and others) for the Replicator 2, as well as criticized for abandoning open-source solutions. From a tech-transfer point of view, MakerBot's evolving business model raises intriguing questions which are being played out now.

Interested in more information? See ***Make* magazine's *Ultimate Guide to 3D Printing*** ($9.99) for a comprehensive survey of the state-of-the-art as of December 2012. This downloadable PDF includes a wealth of information on other 3D printers, hardware, software, materials, tips, and techniques.

If *(when)* you get stuck, MakerBot's online and phone support is helpful, but not always available. Be sure to visit the MakerBot Operators group, the independent 3D Printer Tips, Tricks and Reviews group and the new WIKI at Makerbot.wikia.com.

Tremendous know-how, assistance, and solutions from users are just a click away.

PURPOSE:

The purpose of this guide is to assist the new user in getting the most out of their Replicator 2. While some advanced concepts are mentioned (either out of necessity, or as avenues for further exploration), the main goal is to help new users—who may have no prior 3D modeling or printing experience—successfully explore the exciting world of 3D printing. Experienced users may wish to skip the more introductory content covered in the ZERØ TO HERO sections.

Knowledge about the Replicator 2 grows daily as experience is rapidly gained by users. As a result, communicating ALL current information in one place is inherently a fast-moving target; yet the need for a reference guide for new users (even if incomplete) may be handy.

Feedback and suggestions for improvements are very much welcomed. Drop DigiFabLab a line and let us know if this resource helped and/or how it can be improved.

David Celento, Founder

DigiFabLab
info@DigiFabLab.com

TABLE OF CONTENTS:

» OVERVIEW OF WORKFLOW 6
3 Step Process for 3D Printing: 6
Tips for 3D Digital Model Making: 6
» ZERØ TO HERO: MAKERWARE SOFTWARE DETAILS 7
About MakerWare Software: 7
Details—Size, Format, & Units: 7
Launch MakerWare: 8
Left Buttons: Navigation & Positioning 9
Scale (S): 9
Turn (T): 10
Move (M): 11
Look (L): 12
Camera Home: 12
Top Buttons: Add, Make IT, & Save 13
Add: 13
Make It: Basic Settings 14
Make It: Advanced Settings 17
Export: 19
Save: 19
» ZERØ TO HERO: MAKERBOT REPLICATOR 2 DETAILS 20

Turn on the Replicator 2: 20
Navigating the Interface: 20
Orienting and Preparing the Acrylic Build Plate: 21
Leveling the Build Plate: 21
Unload / Load Filament: 22
Fan Orientation: 25
Printing: 25

» TROUBLESHOOTING: FIXES FOR A FEW COMMON PROBLEMS 28

Poor Adhesion and Failures: 28
Shrinkage, Lifting & Curling Problems: 30
Stripping: 31
Filament Jams: 32
Clogged Nozzle: 34
Squeaking or Screeching: 34

» INITIAL SETUP: RECOMMENDED BEFORE YOU START PRINTING 35

Check All Fasteners: 35
Remove Spring Pin Holding Cables for X Drive: 35
Proper Cooling Fan Orientation: 36
Prepping the Spool Holder & Spools: 36
Oiling the Filament Feed Tube: 37
Proper Drive Wheel Adjustment: 37

» MODS: OPTIMIZING AND HOT-RODDING YOUR REPLICATOR 2 38

Extruder Upgrade: **AN ESSENTIAL MOD** 38
Glass Build Plate: 39
Sailfish Firmware: 40
Aluminum Table Arms: 40

» TOOLS & SUPPLIES, SOFTWARE, AND RESOURCES: 41

Recommended Tools & Supplies: 41
A Software Sampling: 41
A Few Resources: 42

» FAQ's: FROM MAKERBOT'S WEBSITE 43

My build is the wrong size: 43
My Replicator 2 can't see my SD card: 43
I can't find my file on the SD card: 43
Help! My bot sounds like it's shaking itself to pieces!: 43
How do I update firmware on my Replicator 2?: 43
Do I need to do any maintenance on my Replicator 2?: 43

» ACKNOWLEDGEMENTS, ABOUT DIGIFABLAB, & SHARING: 44

Acknowledgements: 44
About DigiFabLab:

info@DigiFabLab.com 44

Use of this Material: 44

» OVERVIEW OF WORKFLOW

All 3D printing processes use these basic steps. Details for MakerWare are shown here.

1. **NOVICE:** Start by printing an example file included with MakerWare.
2. **NOVICE:** Download an existing object from **Thingiverse.com** — an amazing collection of free 3D models designed and submitted by enthusiasts.
3. **ADVANCED:** Design and print your own digital models; thus improving your design (and modeling) skills. **Note that unique designs require very careful modeling!**

3 Step Process for 3D Printing:

- **1: Use a Valid 3D Digital Model**
  - **NOVICE:** Open MakerWare {Free download **here**}, File » Example » {Choose desired file}
  - **NOVICE:** Go to Thingiverse.com » {Download desired file} » {Open in MakerWare}.
  - **ADVANCED:** Create a 3D digital model of your own design.
  - Use any 3D software which exports .STL or .OBJ files.
  - See '**Tips for 3D Digital Model Making**' in next column.
- **2: Tooling the Model - Software:**
  - Position, Orient, and Scale the model in MakerWare.
  - Use 'Make-It' to 'slice' model into layers and generate toolpaths (known as G-code).
  - Save an .X3G to SD card. *Direct connections are also possible, but are more prone to failures.*
- **3: Printing the Model - Hardware:**
  - Select .X3G file on SD card .
  - Level and prepare the print bed.
  - Load the extrusion filament.
  - Print the file.

Tips for 3D Digital Model Making:

- **Polygon Format Needed:** If created in NURBS, or Sub-Divisions, you must convert model to Polygons.
- **Valid Model Characteristics:** Use **Netfabb** (free), **Rhinoceros** (Windows trial, OSX free) to check.
  - **Watertight:** No gaps in model.
  - **Face Normals:** Same direction.
  - **Manifold Edges:** Edges do not share three polygon faces.
- **Thickness:** Desired wall thicknesses should be constructed digitally. If not, the model will print as a solid.
- **File Format:** Export as .STL (binary).
- **Rhinoceros Repair & Test Methods:**
  - **Watertight**: Analyze » Edge Tools » Show Edges » Naked Edges. Repair gaps if found.
  - **Normals:** Type 'Dir' to check. Click to flip faces. Or Type 'Flip'.
    - *Windows recommended feature to color face normals: File » Properties » View » Display Modes » Shaded. {On right side} » Backface Settings » Single Color for all Backface Settings » {Select a color}.*
  - **Convert to Polygons:** Mesh » From NURBS object. Higher counts = accuracy = large files.

» ZERØ TO HERO: MAKERWARE SOFTWARE DETAILS

*This section contains detailed instructions to prepare ("tool" or "slice") a 3D model for printing using MakerWare software (which creates G-code for the printer). Other "slicers" exist (such as ReplicatorG—which is very popular with advanced users—Slic3r, and Repetier-Host) but are not covered here. The model must be '**valid**'. See '**Tips for 3D Digital Model Making**' above.*

1. About MakerWare[1] Software:

- **SLICER:** The primary function of this software is to 'slice' your model per the desired layer thickness, then generate toolpaths (or 'G code' instructions) for the printer to follow. For this reason, 3D printer software packages are generically referred to as 'Slicers'.
- **OS:** Windows, Mac, and Ubuntu Linux available.
- **FREE DOWNLOAD:** If you install MakerWare, you can tool files on your own computer, then transfer them to the MakerBot printer via an SD card. This is very convenient since you don't have to be hooked up to the printer while printing. While MakerWare will print directly from a computer to the printer (via a USB cable) this is not recommended since the computer is tied up and the network connection is easily interrupted.
- **Download MakerWare Here**[2]
- **PROS:** Fast (Low and Medium settings) and Easy. Customizable. Pretty good slicing routines for most basic jobs.
- **CONS:**
  - The slicing algorithm for the MakerBot Slicer (used for the Low and Medium options) instructs the head to continue to extrude while "traveling" creating diagonal lines in the print layer. These lines may be objectionable or render intricate prints unusable.
  - Advanced control is not easily done as a few parameters are less developed in the customizable scripting profiles than are possible using Skeinforge in ReplicatorG.

2. Details—Size, Format, & Units:

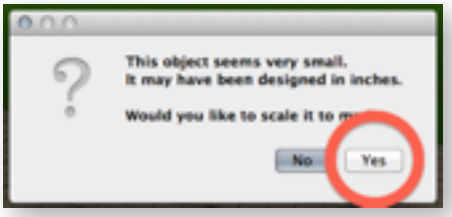
- **LAYER THICKNESS HEIGHTS:** *(Note that the extruder tip is 0.4 mm wide.)*
  - The **Low** default height (0.34 mm) is very fast, but layers are more visible. Variables can be customized.
  - The **Medium** default height (0.27 mm) **is recommended** for most prints, balancing speed, finish, and reliability nicely. Variables can be customized.
  - The **High** default height (0.10 mm) uses a different slicing engine (Skeinforge) which is much slower to generate. Prints done at the default resolution often fail with the stock MBI extruder (a delrin plunger). Variables can be customized and setting the layer thickness to **0.15 mm** seems more reliable if finer prints are desired than Medium.

---

[1] *Tutorial based on MakerWare Beta v 1.1 with updates from v 2.0 included. Check for frequent MakerWare Beta updates!*

[2] *http://www.makerbot.com/makerware/%23downloads*

- You may fine-tune various parameters under Advanced Options.
- **MAX SIZE:** The maximum build envelope is 285mm L x 153mm W x 155mm H *[11.2" L x 6.0" W x 6.1" H]*
- **FORMAT:** Files must be in .STL or .OBJ format and 'valid'. *[See prior instructions]*
- **UNITS:** MakerWare works only in millimeters (mm). If you add a file that is in inches, you may be prompted to convert it to mm.



  - If not prompted, you can convert inches to mm as follows: {Select object} »Click the Scale button » Click the Triangle » Change Dimensions » Inches --> mm

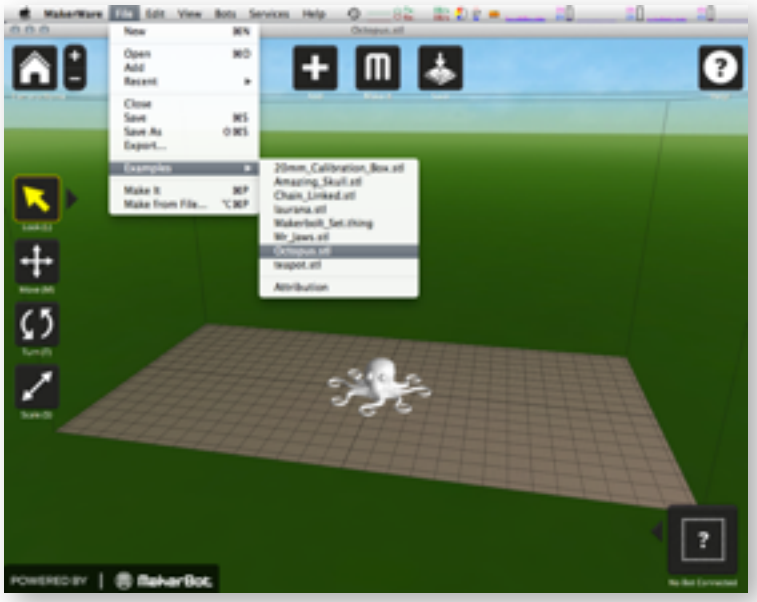
3. Launch MakerWare:

- **Objects You can print:**
  - Examples included with MakerWare.
  - Designs by others. *[Check out **Thingiverse.com**, and download something].*
  - Your own designs. *[Consider sharing your work on **Thingiverse.com**].*

- **Learning the interface:**
  - **Load an Example File:** File » Examples » {Select desired item}

  or

  - **Load Your Own .STL or .OBJ File:** File » Open, or Click the Add button.

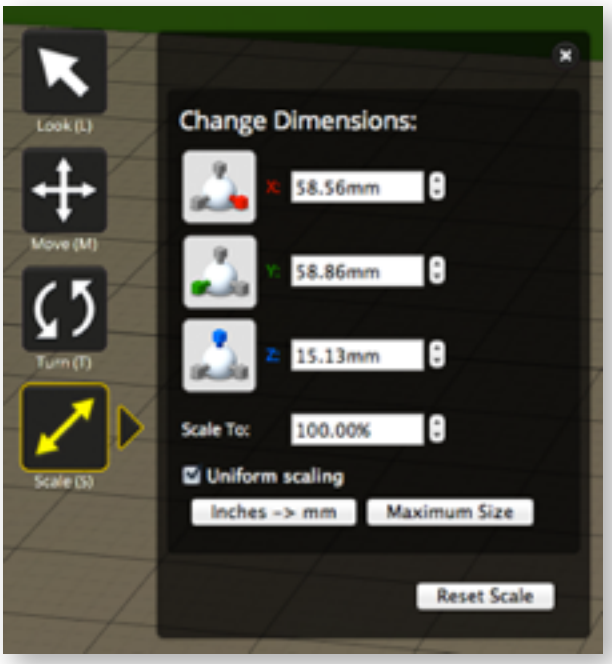


LEFT BUTTONS: NAVIGATION & POSITIONING

4. Scale (S):

With your object selected, click the Scale button, then click-drag your mouse to scale your object up or down (units shown in millimeters). Click the Triangle next to the button (or double-click the button), to open the Change Dimensions options window. 'S' is the Hotkey.

- **Inches -> mm:** Scale inches to mm if you did not do this when loading your file.[3] *(You need to do this if your object looks teeny weeny).*
- **Uniform Scaling:** Likely you want to leave this checked unless you're into distortion.
- **X,Y, Z:** Easily set a known or desired dimension in mm. If **Uniform Scaling** is selected, the result will be as predicted. If not, distortion ahead!
  - *X is the LONG direction, Y is the SHORT direction, and Z is the VERTICAL direction.*
- **Maximum Size:** This fills the entire print bed. Your print will be gigantic, will take forever to print (and may fail), use a lot of material, and is NOT RECOMMENDED!
- **Scale To:** Easy to dial in percentages. *[NOTE: The percentage readout does not change with arrow input, but object size changes.]*
- **Reset Scale:** Always useful if you mess things up. Takes you back to the original size.

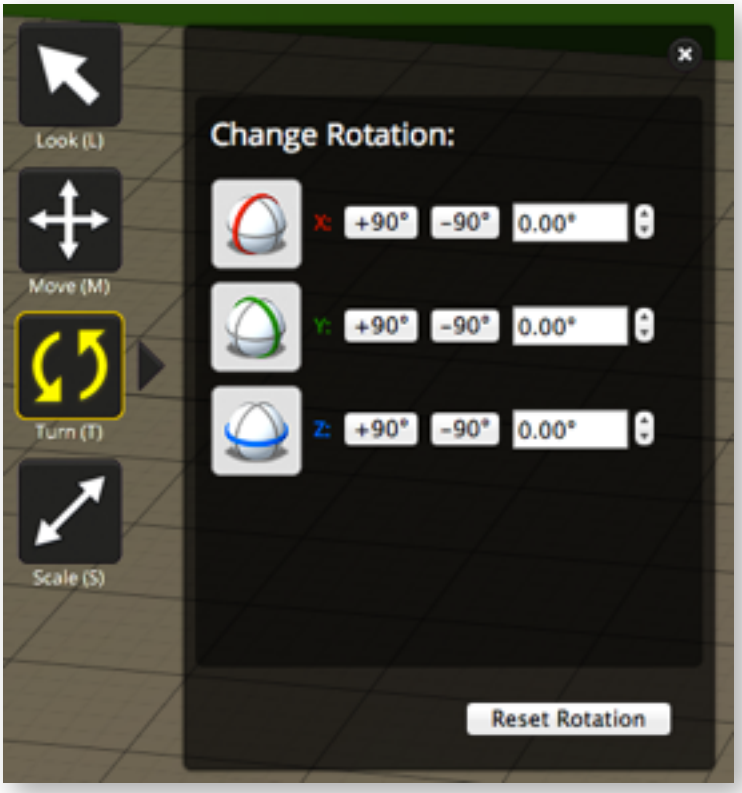


[3] *Feature Request: A preference that permits working in inches would be welcomed by U.S. users.*

5. Turn (T):

With your object selected, click the Turn button, then click-drag your mouse to rotate your object in degrees. Click the Triangle next to the button (or double-click the button) to open the Change Rotation options window. This is similar to the Rotate command in many other programs. 'T' is the Hotkey.

- **X, Y, Z:** You can rotate around the X, Y, or Z axes.
  - The + 90° and - 90° are useful for positioning your model in the optimal orientation, minimizing, or eliminating, the need for support material.
- **Reset Rotation:** Takes you back to your original orientation.



6. Move (M):

With your object selected, click the Move button, then click-drag your mouse to move it. Coordinates will be displayed in relevant axes. Click the Triangle next to the button (or double-click the button) to open the Change Position options window. 'M' is the Hotkey.

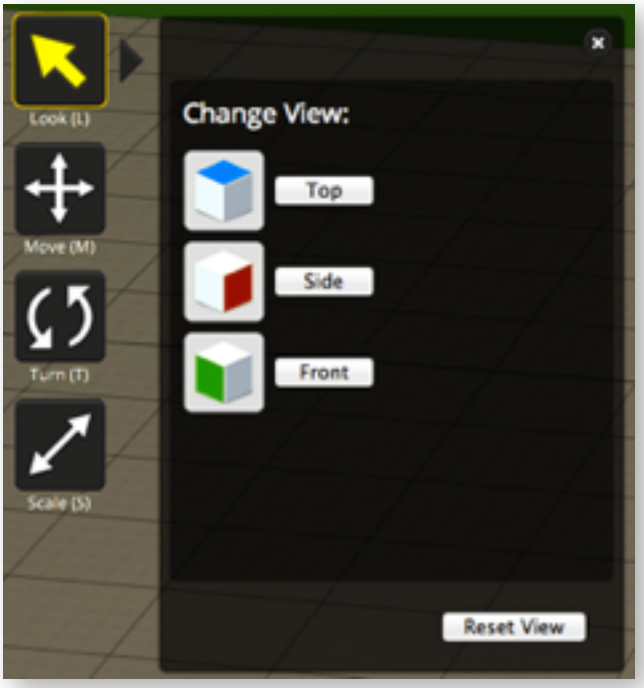
- **On Platform: **It is CRITICAL that this be Clicked.****
  - *If an object is below the platform, tests indicate that the MakerBot print head may crash into the table.* [4]
- **Center:** This is a convenient way to put your model in the center of the platform. Tests indicate that this is not always accurate, but it is helpful.
- **X, Y, Z:** An obvious mechanism to move your model about.
  - It is unclear why one would use the Z adjustment, since this would raise or lower the object off of the platform. *Be careful with this!*
  - If you move your object up in the Z direction, this creates additional support. In rare cases this may be useful such as for tightly nesting multiple parts?
  - Down in the Z direction moves your object below the platform and will likely crash the print head. *Avoid this. Really.*
- **Reset Position:** Restore your original position if you wish.
- **NOTE:** If your object is not oriented to be flat on the bottom when you export it, MakerWare will not automatically position it flat on the build plate. Instead, it will simply place the lowest point on the build plate, regardless whether it is flat or not.

[4] *Feature Request: Ideally, MakerWare would not recognize anything below the table and slice only what is above the table — this would be convenient for quick tooling and partial test prints using various settings.*

7. Look (L):

Click the Look button, then click-drag your mouse to change your view. Click the Triangle next to the button (or double-click the button) to get the Change View options.[5] 'L' is the Hotkey.

- **Top, Side, Front:** These are self explanatory
- **Reset View:** Returns to Perspective (or Camera Home)
- **Pan:** *Shift + Left Click-Drag* (Mouse) or Middle Mouse Button (MMB). *Shift + Alt + Click-Drag* (Trackpad)
- **Rotate:** *Rt Click-Drag* (Mouse). *Alt + Click-Drag* (Trackpad)
- **Zoom:** *Scroll Wheel* (Mouse). *Two Fingers* (Trackpad)



8. Camera Home:

Simply restores the default perspective view. The + and - buttons zoom in and out.

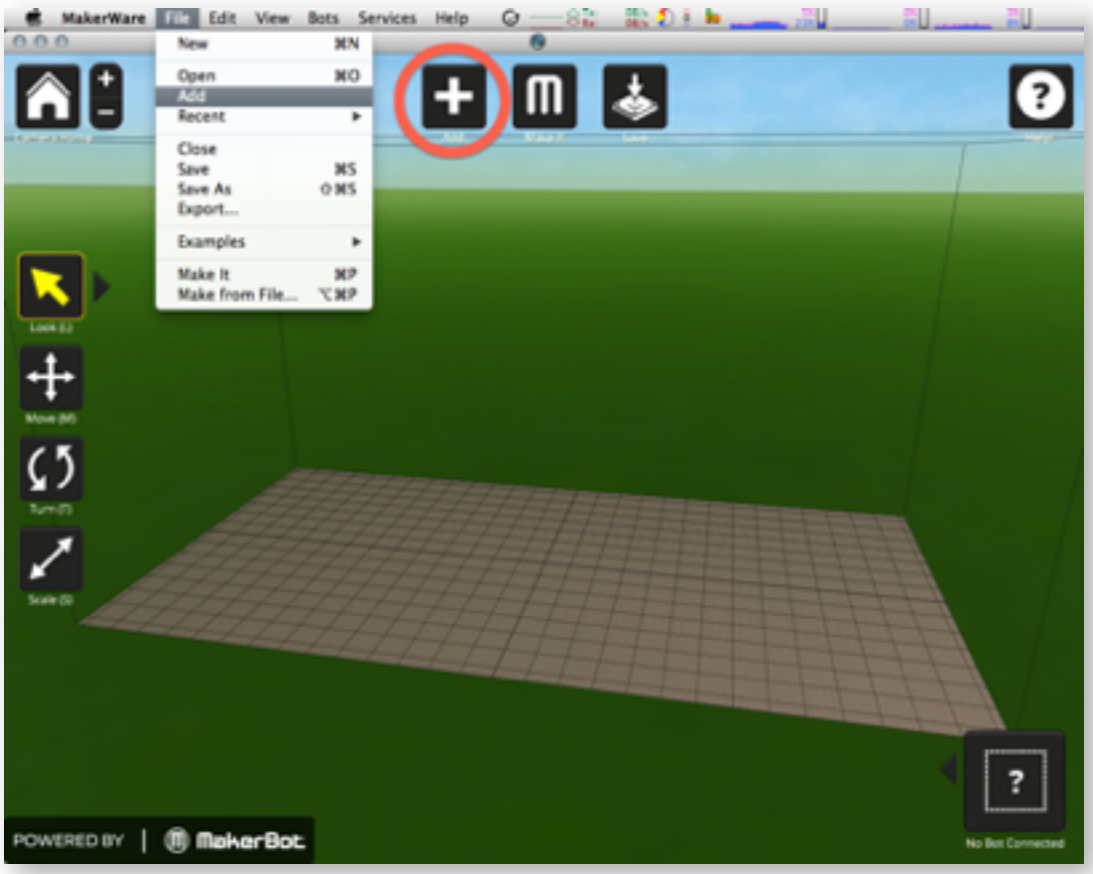
[5] *Feature Request: It is difficult to tell the precise location of your object with perspective on. A button that turns off perspective, giving just orthographic information, would be useful to ensure the object fits within the build envelope.*

TOP BUTTONS: ADD, MAKE IT, & SAVE

9. Add:

- **Add your own design or a downloaded file:**
  - Click Add Button » *{Select desired file}*.

    OR
  - File » Add » *{Select desired file}*.
  - Both methods permit multiple parts to be inserted and printed.

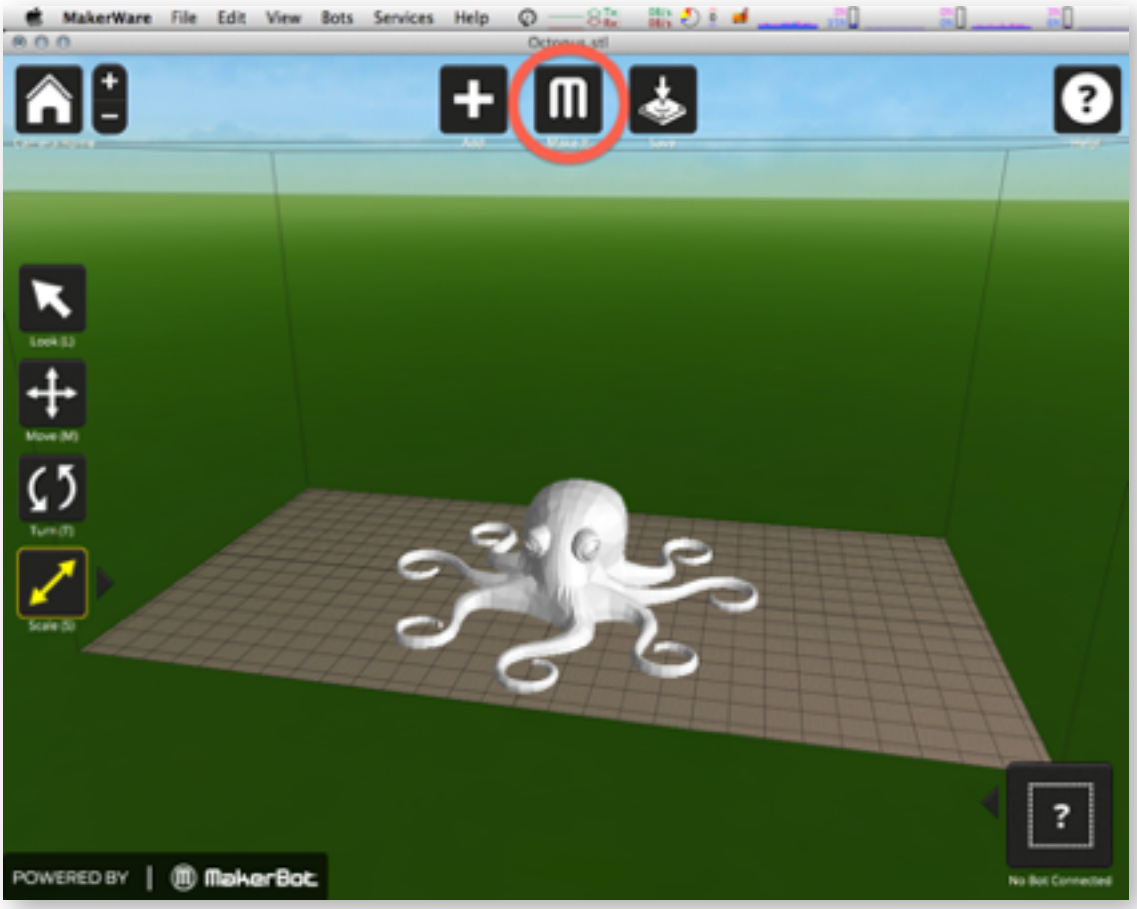


10. Make It: Basic Settings

This is the step where your design is sliced into layers and 'tooling' instructions are made for the printer (an operation which occurs in the background). Tooling is referred to as G code or NC code, which is essentially a text file with instructions for machine movement. This operation will generate a proprietary .X3G file which can be read by the MakerBot.

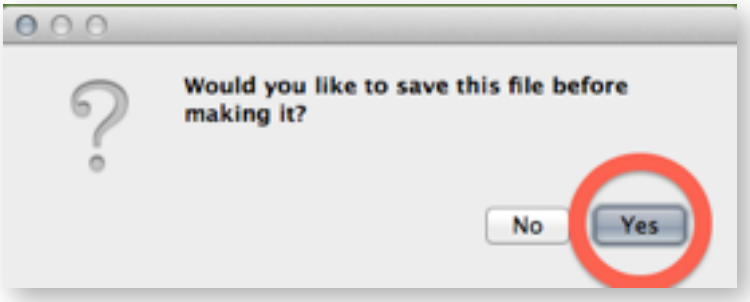
At this time there is no way to view (or edit) the G code, visually preview the tool paths, or estimate time and/or material usage—all of which are areas for future improvements..[6]

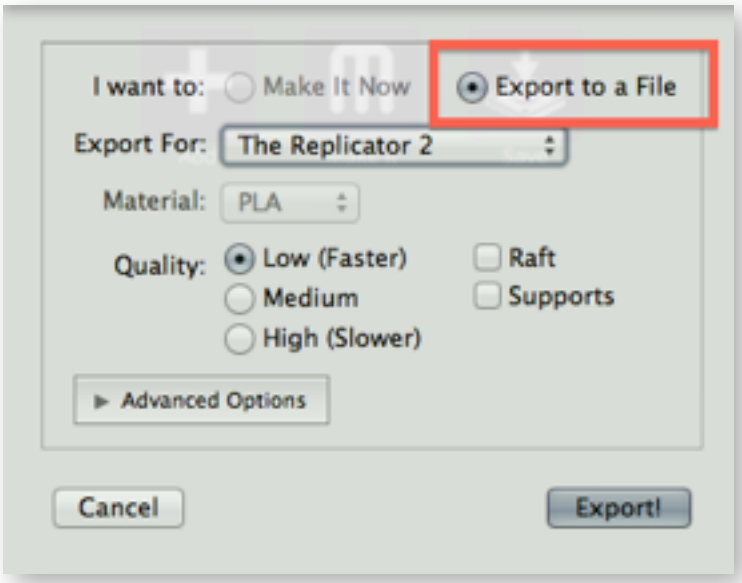
- Click Make It Button



[6] *Feature Request: All of these features would be very useful. Material usage, in particular, is a critical calculation for environments where printers are shared and material usage fees are charged to users.*

- You may be prompted to Save » Select Yes to create a .Thing file.
  - .Thing is actually a sort of .zip file which saves your .stl and a separate .json file. If you lose your .stl file, but have your .Thing file, you can rename the extension and open a folder which contains both.





- **I want to:**
  - **Export to a File**—*Preferred*. This saves the tooled file in .X3G format. The file can be put onto an SD card and printed without the MakerBot being hooked up to a computer the entire time it is printing.
    - Note that the SD card can be no larger than 2G and must be formatted as FAT16.
    - Recommended is that you also save a .gcode file. While you don't need this to print, you may use other applications to view the slicing toolpaths to identify any problems before you print, such as Repetier-Host and Pleasant 3D (Mac only).
  - **Make it Now**—*Not Preferred.* Choose only if the computer will be hooked up to the MakerBot for the duration of the print. If using this method, with the MakerBot connected to the computer via the USB cable and turned on, you must establish a connection by doing the following: MakerBots » Type of MakerBot » Replicator 2. Also Services » Restart Background Service if it is not recognizing the printer.

- **Export For:** The Replicator 2
- **Material:** PLA is the material of choice for Rep2. *[ABS requires a heated build plate]*
  - Some users have reported good results printing with Nylon (with a wooden or bamboo plate, higher heat, and with no cooling fan), PVA, and HIPS (both of which are soluble materials).
- **Quality:**
  - **Low** (Faster)—Fastest option. Roughest finish. Layer height is 0.34 mm
  - **Medium**—***Recommended***. Good balance between speed and finish. Layer height is 0.27 mm. Troubleshooting is best done with this setting.
  - **High** (Slower)—Slowest option. Smoothest finish. Layer height is 0.10 mm. Note that this option uses a different slicing engine (Skeinforge) and is much more prone to failure during larger prints. Tests have shown that tweaking the thickness to 0.15 mm in the Advanced Options creates more reliable prints of similar finish and in less time. The build plate must be flat and nozzle clearance must be precise.
- **Raft:** This puts your object on a printed bed of support material, slightly above the plate. You will have to remove this support material, compromising the finish on the bottom.
  - Useful if your object is fragile or not flat on the bottom.
  - Often this improves adhesion of the model to the plate.
- **Supports:** [7]
  - Supports are a less dense grid of PLA to support horizontal overhangs or any elements which are at a shallower angle than 45° to the build plate.
  - You will have to remove this support material and this can be *very* time-consuming (and frustrating), compromising the finish. At this time, it is best to minimize or eliminate the need for supports when designing and/or orienting a part, if at all possible.
  - Enable Supports if your object has overhangs or low angles. By default, you will also have to enable Raft (which is unfortunate).
  - If you have conditions which require support and you do not enable Supports, your print may 'droop', or fail, in unsupported areas.

[7] *Note that some dual head printers—such as the new Replicator 2X—permit dissolvable support material made from PVA. This capability would be very useful; however, software slicing to create separate supports is not yet integrated in MakerWare. Further, PVA has not yet been widely adopted. DigiFabLab is not yet able to offer guidance using this material.*

11. Make It: Advanced Settings

It is worth playing with the various options here and customizing them to your liking. Tweaking the Advanced Settings —especially Infill, Shells, Layer Height, and Temperature —can go a long way toward delivering optimal results.[8]

One can even dive into advanced coding through the Create Profiles[9] button—which is not for the meek! See **here**.[10]

Once you gain confidence with MakerWare, you may wish to experiment with other open-source slicing programs, such as ReplicatorG+Sailfish (a popular slicer which offers a high degree of control and works very well with Sailfish firmware installed on your Replicator 2). You can get it **here**.[11] Repetier-Host, KISS, Cura, and Slic3r are a few others of possible interest.

- **Quality:** This displays the Default values being used
  - **Infill %**—Refers to the density of infill inside a solid part.
    - 0% means no infill. Print will be hollow.
    - 100% means the print will be entirely solid.

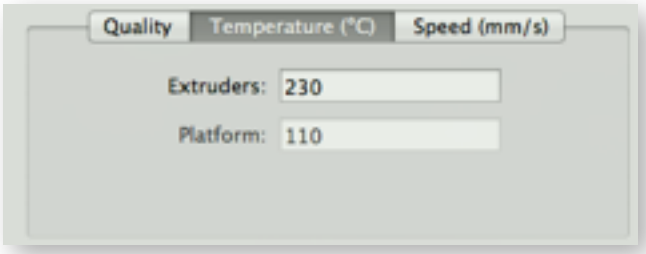
[8] *Feature Request: For SD card users, the ability to select a bot and save modified profiles when not hooked to a bot.*

[9] *Feature Request: The ability to edit the Default Profiles in Create Profile, to make slight adjustments.*

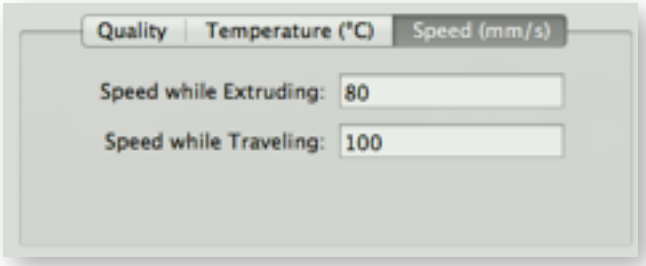
[10] *http://www.makerbot.com/support/makerware/documentation/advanced/*

[11] *http://www.thingiverse.com/thing:32084*

  - Infill between these numbers is an eggcrate of the assigned density. It is stronger than you think, will save material, decrease print time, and help reduce curling.
- **Number of Shells**—Refers to the number of passes which make up your wall thickness. Zero means only one skin. The higher the number, the thicker the 'skin', the stronger the surface.
  - If you encounter curling problems, try reducing the shell numbers.
- **Layer Height**—This number defines the smoothness of the finished product in the Z dimension. Lower numbers result in thinner layers and longer print times. Higher numbers result in thicker layers and shorter print times.
- The High setting uses a different slicing engine (Skeinforge) than Low or Medium (which uses MakerBot Slicer). Processing is slower and toolpath outcomes may vary significantly as a result. One benefit is that the diagonal extrusions while traveling (from MakerBot Slicer) are eliminated.



- **Temperature (°C):** Temperature of the extruder head which melts the filament.
  - **Extruders**—The default is 230°C (for all default settings).
    - Worth noting is that RepRap.org indicate melting temperatures from 180°C to 220°C; thus if you are having problems with curling, lowering the temperature may be worth experimenting with. As one lowers the temperature adhesion between layers begins to reduce until it fails entirely.
    - DigiFabLab has found reliable extrusions between 210°-220°C. The lower the temperature, the less glossy the print is.
  - **Platform**—While indicating 110°C, this item is greyed out since the platform on the Replicator 2 is not heated.
    - Heated beds are not essential for printing with PLA (but are for ABS printing on other machines). Despite this, a heated bed may be beneficial to help reduce lifting and improve adhesion with PLA. Keep an eye out for a possible future upgrade from MBI.



- **Speed (mm/s):**
  - **Speed while Extruding**—The rate of the extruder head while printing.
  - **Speed while Traveling**—The rate of the extruder head while transitioning between printing locations. While shown as 100 on Medium, the other default settings indicate 150. Tests show that changing this to 150 seems to work fine, decreasing print time.

12. Export:

- This will make an .X3G File that contains the tooling information.
  - Copy this file onto the SD card to insert into the printer.
  - Note that the SD card can be no larger than 2G and must be formatted as FAT16.
- As noted earlier, it is also worth saving a .gcode file if you wish to pre-visualize slicing toolpaths in another program before printing the file. Power users may also edit the .gcode (which is simply a text file).

13. Save:

- Before quitting, save your file (in .Thing format) in case you need to perform any editing later. As noted earlier, this is essentially a .zip file with a different extension, containing your .stl file and a text file with instructions for MakerWare. You can access these files by changing the extension to .zip.

» ZERØ TO HERO: MAKERBOT REPLICATOR 2 DETAILS

This section contains detailed instructions on printing with the Replicator 2. Basic introductory information is included, as well as a few more detailed topics. No prior experience is needed.

» IMPORTANT «

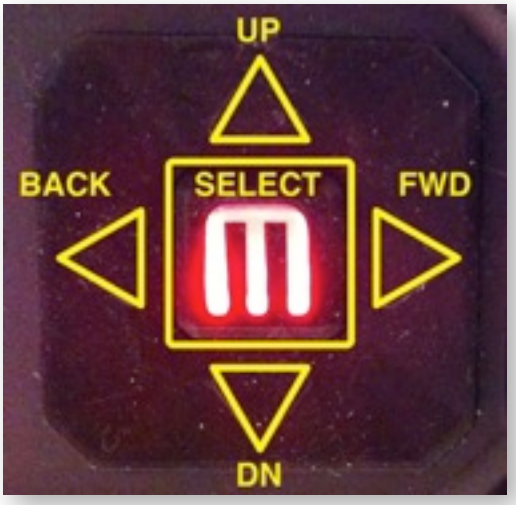
The INITIAL SETUP section should be reviewed before using your machine for the first time to ensure optimal printing results!

1. Turn on the Replicator 2:

- Power switch on rear.
- Current firmware should be displayed on front.[12]
  - If prompted to update firmware, in MakerWare, follow instructions for download.
  - Connect computer to MakerBot via the provided USB cable.
  - MakerWare Menu » MakerBots » Upload Firmware

2. Navigating the Interface:

- **Blinking M:** Waiting for Input.
- **Solid M:** Performing Work.
- **Arrow Buttons:** Performs functions as described in image below.



[12] *Current MakerBot firmware for Replicator 2 is version 7.2 as of this writing. Experienced users may wish to upgrade to the more fully featured and transforming **Sailfish Firmware**, from Jetty Firmware. http://www.thingiverse.com/thing:32084*

3. Orienting and Preparing the Acrylic Build Plate:

- **Which Face to Print On?:**[13]
  - **Smooth Face w/ Logos:** This surface provides the strongest adhesion of the print to the plate if the logos are covered. While difficult to remove the print, this appears to be the best option for prints with large surface area or if you're experiencing adhesion problems.
  - **Frosted Face:** This surface provides weaker adhesion. If the print is small and the first few layers stick, this permits the easiest removal of finished prints.
  - **Blue Painter's Tape:** A number of users swear by blue painters tape by 3M (#2090) covering the top of the build plate. Tests at DigiFabLab produced mixed results. Users are cautioned to not leave prints unattended if using tape. If the print gets dislodged, the tape may force the print into the nozzle causing it to ball up on the head.
    - Use a plastic spreader to firmly apply the tape, without gaps.
    - Be sure to re-adjust the plate height afterwards by re-leveling it.
- **Glass Plate:** DigiFabLab recommends that users install a glass plate (with a thick coat of AquaNet hairspray on it) since very few MBI acrylic plates appear to be entirely flat. *[See the MODS section for details.]*
- **Removal and Installation of Plate:**
  - **Removal:** The plate is removed by pushing the black tab towards the rear to release the plate. Angle the plate out of notches and lift free of the table.
  - **Installation**: Installation is the reverse of removal.
  - **Shimming the Plate:** Tests have shown greatly improved results when shimming the table so the plate firmly locks into the notches. This increases stability while printing.
- **Cleaning the Surface - VERY IMPORTANT!!!**
  - **Acrylic Plate:** Use a razor scraper to remove residue. Isopropyl alcohol (91%) should then be applied with a paper towel to the entire surface to remove fingerprints, dust, etc.
  - **Blue Painter's Tape:** If using blue painter's tape, some users report success lightly sanding the surface with 400 grit sandpaper, then wiping down with alcohol.

4. Leveling the Build Plate:

It is **CRITICAL** that the build plate be trammed (which MBI calls "Leveling") and properly adjusted so that prints adhere properly. Improper adjustment results in poor adhesion of initial layers and failed prints. Tests have shown that a tighter gap between the nozzle and the plate (0.10 mm) provides better adhesion. If one hears 'clicking' sounds or the nozzle drags on the plate while printing, the gap is too tight.

[13] *The stock acrylic plate is rarely completely flat. This is a significant problem when trying to print at resolutions below 0.27 mm. An entirely flat surface is critical for all prints, but especially so for fine resolution prints. Many have upgraded this component with great success to glass, using Aqua Net hairspray after cleaning. See MODS section for details.*

The Run Startup sequence below offers a good overview of *adjusting the build plate. This is highly recommended for the first-time user.*

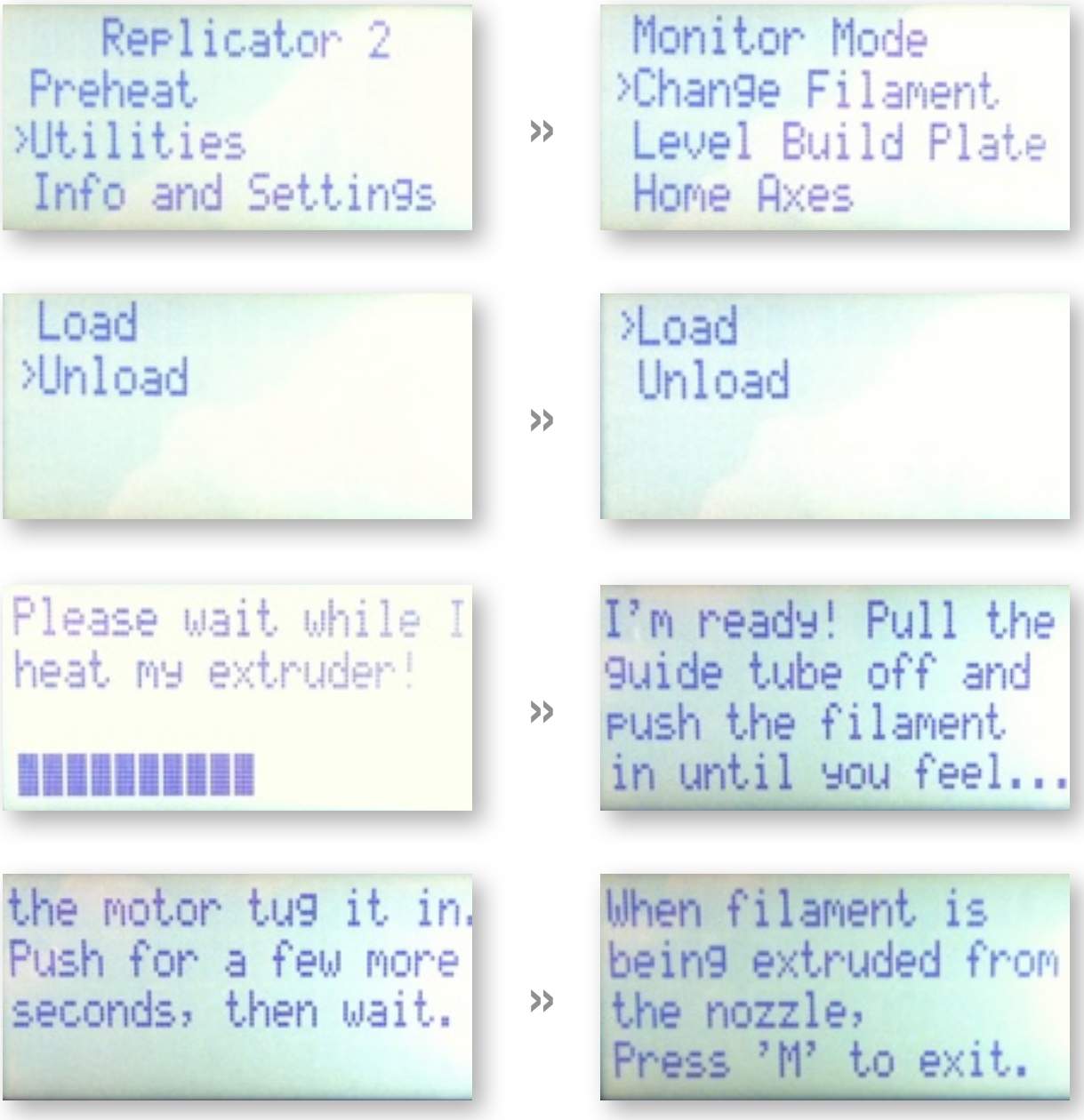
A number of self-explanatory steps follow which are not shown…

- **Utilities** » **Level Build Plate.** This routine will ask you to adjust the gap under the extrusion nozzle at various points on the build plate, adjusting the three leveling screws on the bottom of the table.
  - A 0.10 mm (0.004") feeler gauge (or a crisp U.S. $1 bill) is recommended. You should adjust the screws, starting at the rear center, so you feel slight drag under the nozzle.
  - You will likely find that the plastic build plate is slightly warped and not entirely flat throughout; thus, the outer point adjustments may not match the inner point adjustments. The most critical points to adjust accurately are the points above the leveling screws (rear center first, front mid-right, front mid-left) for small prints that are centered on the plate.
  - If the stock plate is warped, exiting from this routine after the first three points are adjusted (left button on control panel), then re-running the script to verify these three points, is the most useful approach for smaller prints that are centered.
    - Note that if you are attempting large prints and the build plate is not flat, you will likely have adhesion problems and possible failures.
  - Non-intuitively, turning the screws on the bottom of the table CW (clockwise) makes the gap bigger. CCW (counter-clockwise) makes the gap smaller.
  - Some users report significant printing improvements after replacing the build plate with glass to correct the warping of the plastic plate—an upgrade well worth considering for large prints, or prints finer than 0.27 mm height. *[See MODS section]*

5. Unload / Load Filament:

Filament should unroll from the bottom of the spool and extend up into the tube. If using the left holder (looking at the back), the spool should unwind counter-clockwise. If using the right holder, the spool should unwind clockwise.

- Be sure to keep spools free of dirt and particulates to prevent nozzle clogging!
- Be sure that the inside of the spool is completely smooth. Sand if not.
- If you have not slightly filed down the top edges of your spool holder, this is also advised.



- **Vegetable Oil:** A number of users have reported success rubbing a few drops of vegetable oil on the end of a section of filament, then "flossing" the protective tube. This seems to reduce stiction between the filament and the tube, allowing the filament to be pulled more freely by the drive wheel.
- **Loading Tips:** During Loading, remove plastic tube on top, trim filament on a diagonal, and gently assist the filament when feeding it in. The drive wheel should catch the filament and begin extruding. If you have the stock extruder (a delrin plunger), you are advised to visit the MakerBot website that addresses proper adjustment, **here**.[14]

[14] *http://www.makerbot.com/support/replicator2/documentation/maintenance/*

- DigiFabLab does not recommend use of the Replicator 2 with this OEM part due to a number of challenges with proper adjustment and inconsistent filament pressure.

- **If Using an Upgraded Extruder:** This section applies only to machines with the upgraded extruder mechanism by **whpthomas**[15], or **MakerBot,**[16] etc. This upgrade is strongly recommended for all Replicator 2 machines, as soon as possible.
  - Depress the thumb lever only enough to insert the filament between the drive wheel and the pressure bearing. Once inserted and advancing, release the thumb lever for the filament to advance on its own.
  - The pressure on the filament is now automatically regulated by the spring and no longer needs to be adjusted.
  - You should be able to squeeze the filament with your fingers, but not stop it. If you can stop the filament, your spring is too weak.
  - If you hear "clicking" from the drive wheel, your spring is either too tight or your extruder is binding and needs to be loosened, sanded, and/or lubricated. If warped, you must replace it.
  - You may print the major parts for the upgraded extruder yourself. You will need to purchase the spring, screw, and bearing listed on the Thingiverse.com sites, or directly from the MakerBot store, if using their design. *[See MODS for details.]*
  - Instead of fabricating this upgrade, users may purchase an upgraded extruder assembly from MBI.
  - **NOTE:** PLA warps with heat, thus printing an assembly on a Rep 2 in PLA may only provide a semi-temporary solution, requiring replacement every now and then. If one can get these parts printed in ABS (from MakerBot or a friend), these do not seem to warp with heat and are more durable.

- **If Using the Stock Extruder (Delrin Plunger):** It is likely that you will need to adjust the delrin plunger mechanism to apply proper pressure to the filament before, during, and/or after loading. This requires removing the lower cooling fan, is time consuming, and will be short-lived since heat changes the o-ring's properties.
  - Be VERY careful to not snap the wires off of the fan! Routing them under the retainer clip, and placing tape over them is advised.
  - If you hear 'clicking', or can not get the filament to contact the drive wheel, the plunger pressure is too tight.
  - If you can squeeze the filament with your fingers and stop the extrusion, or if the extrusion is not uniform, the plunger pressure is likely too light.
  - MBI reviews methods to adjust the stock plunger mechanism **here**[17] and has posted a video **here**.[18] *[Good luck -- especially with prints finer than 0.27 mm layer height. You'll soon understand why MBI copied whpthomas' excellent solution, and why you should upgrade this part.]*

---

[15] http://www.thingiverse.com/thing:42250

[16] http://www.thingiverse.com/thing:53125

17 http://www.makerbot.com/support/replicator2/documentation/maintenance/

18 http://www.youtube.com/watch?v=QOJN_8AAC9U

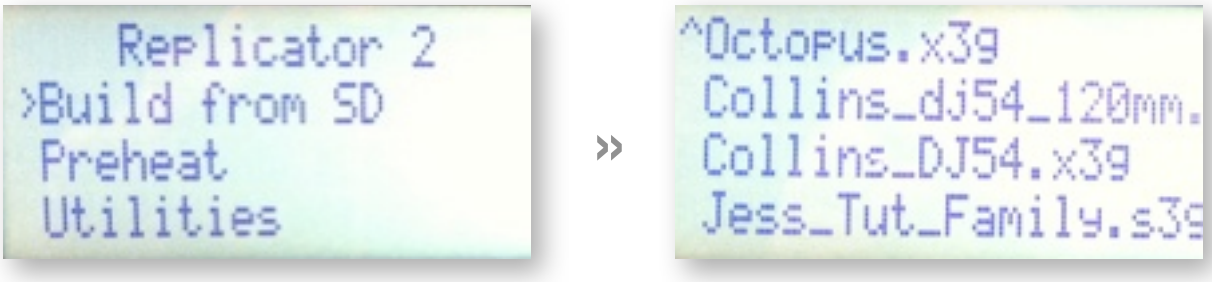
- **Filament Not Advancing?:** If filament does not advance and start extruding from the nozzle, you likely have a jam. The only solution is to disassemble the extruder head to manually clear the jam. *[See the TROUBLESHOOTING section for details]*
- **Straight Extrusion:** Filament should extrude straight down and uniformly. If your filament is tilting to one side, or curling as it comes down, you likely have a particle jammed in your nozzle. Only disassembly and cleaning or replacement of the nozzle will correct this.
  - MakerBot offers instructions **here**[19] for a similar unit.
  - Be sure that nozzle is hot before trying to remove it, or you will snap the nozzle.

6. Fan Orientation:

- **Blowing IN not OUT:** While loading filament, the fan on the front of the extruder head should be spinning, to cool the heat sink. Verify that this fan is blowing IN towards the heat sink, not out, away from the heat sink.
- If you can read the label on the front of the fan, reverse the fan to blowing in. This will improve cooling of heat sink. *[Turn the bot off to do this!]*

7. Printing:

- Insert SD Card (slot on top, behind LCD screen).
- Scroll up » Select 'Build from SD' » {Select File by Scrolling Up and Down} » Enter



[19] http://www.makerbot.com/support/replicator/troubleshooting/nozzle-swap/

- **Extruder:** Value on left is actual head temp, value on right is target temp in celsius.
- **Percentage:** Completion is shown in upper right corner while printing.[20] It is believed that this percentage is based upon the number of layers processed, rather than the percentage of the actual volume of the model printed.
- **Pause/Cancel:** Prints may be paused or cancelled. *[Left arrow shows mid-print options]*
- **Change Filament:** Filament may be changed during mid-print (by pausing). *[Try switching filament colors for interesting results.]*
- A partial Octopus print is shown above with interior scaffolding evident for 10% infill.

8. Removing the Model from the Build Plate:

- **Small Prints:** *Carefully* use a razor blade/scraper for removing small prints.
- **Large Prints:** A wide metal putty knife or craft spatula may help remove large prints.

9. Removing Scaffolding from the Model:

This can be very time-consuming and challenging.

- Needle nose pliers work well for rough work.
- A razor scraper works well for finish work. *Be very careful and **ALWAYS** cut away from you!* Because PLA is very hard, the razor scraper can easily slip off the PLA.
- Detail work and tight crevices may be addressed with dental picks.

10. Finishing the Model:

- Nothing need be done if you like the look of the plastic and the layer lines, as many do.
- If a smooth finish is desired, you will find that PLA is *VERY* tough material. Sanding with 100 grit sandpaper, then working up to 400 grit for a smooth surface is generally the best method to remove flaws or inconsistencies—especially those from a base Raft.

[20] *Feature Request: An option to show print time remaining. Sailfish firmware , a recommended MOD, provides this, and a number of other nice features which improve print results and user experiences. A highly recommended upgrade!*

  - The **MSDS sheet for PLA** suggests that the dust is quite hazardous, recommending the serious measure of a pressurized face mask. With this being a relatively new material, even if organic, *extreme caution is advised* when sanding, presumably because of the molecular structure of the compounds used.
  - Pressurized face masks are extremely expensive, and used most often when working with volatile materials which have hazardous off-gassing fumes. Being that particulates are involved, sanding should, at a minimum, be done only when wearing canister filter style masks, rather than simple fiber-based white dust masks.
- For a very smooth finish, users report good results using several layers of Krylon paint, then wet-sanding (with 400 grit minimum). Using several base coats of Filler Primer will help fill the voids more quickly, prior to painting.

» TROUBLESHOOTING: FIXES FOR A FEW COMMON PROBLEMS

The most common problems using the Replicator2—after upgrading the feed mechanism—are adhesion to the build plate and lifting or curling of the corners of a print. Occasionally, you may also experience stripping and filament jams (both of which are very common problems with the OEM feed mechanism). These issues are defined and addressed below.

1. Poor Adhesion and Failures:

Poor adhesion of initial layers—such as clumps, strings, ropey-ness, peeling, and general inconsistencies—are usually the result of one or more of the following *(listed in descending order of frequency)*: A) a dirty build plate, B) warped build plate, C) a gap that is too large between the build plate and the extruder head, D) incorrect pressure on the filament at the drive wheel, E) too slow a print speed at a fine resolution, G) a clogged nozzle, or H) a head temperature that is either not reporting or heating properly. Poor adhesion may occur at one or more areas, or across the entire build surface.

- **Step 1:** Check the build plate flatness with a long straight edge across the entire surface. Using a dial gauge is even better. Can you see any gaps or slip a thin feeler gauge under the edge anywhere?
  - If so, your build plate is not flat (few stock plates seem to be flat across the entire surface). If you can not add thin glass on top of the plate, or replace the plate with 3/8" thick glass, you will have to position your model where it is most flat.
- **Step 2:** Use the shiny side of your build plate (with the logos on it) for maximum adhesion. Adjust the tramming of the build plate and the gap between head and (the flattest part of) the plate using the leveling function under Utilities on the bot.
  - The use of a Raft often helps improve adhesion.
  - Placing Helper Disks at the corners also helps. File » Examples » Helper Disks.
  - Some users like to use a dial gauge and jog it across the plate. Whpthomas has provided a very nice dial gauge attachment on Thingiverse—**The Snap-on Dial Indicator Holder for Replicator 2.**[21]
  - Whpthomas, once again, offers a nice leveling solution on Thingiverse—the **Interactive Build Plate Levelling - Print Aid**.[22]
- **Step 3:** Clean the build plate with rubbing alcohol before printing. Even fingerprints will reduce adhesion.
- **Step 4:** If you have upgraded your extruder to a spring-loaded version, be sure there is no "stiction" in the lever. It should operate smoothly and freely when it is depressed. If it feels a little "sticky", and you need more force to initiate movement than is needed to sustain motion, you need to disassemble the mechanism and loosen screws, sand parts, and/or lubricate the hinge mechanism. Be sure it is not warped and binding. If so, replace it (ideally with ABS pieces).

---

[21] *The Snap-on Dial Indicator Holder for Replicator 2* **http://www.thingiverse.com/thing:41090**

[22] *Interactive Build Plate Levelling - Print Aid, by whpthomas:* **http://www.thingiverse.com/thing:39430**

- **Step 5:** If you have a stock OEM delrin plunger installed, you need to readjust the pressure before proceeding.[23]
  - Note that the OEM plunger is not well-suited for prints that are less than 0.27mm thick. Finer layers may work if this is perfectly adjusted, but often prints fail later during the print as 'heat soak' begins to occur, changing the pressure applied by the stock plunger.
  - The symptom of a heat-soak failure is "fuzzy" ends on the last few layers, then a phenomenon known as 'air printing', with nothing coming out of the extruder, as if it is blocked, but the extruder continuing to move. Most likely, the filament is stripped. *[Perhaps this is a good time to mention yet again the importance of upgrading the extruder?]*
- **Step 6:** Load a solid color filament for troubleshooting—white if possible.
  - Avoid the clear filament shipped with your machine for troubleshooting. Numerous users report difficulties with this filament due to diameter variations, inflexibility of the material, and possibly increased "stiction"—all of which easily overwhelm even a properly functioning stock delrin plunger.
  - Spring loaded extruders seem to handle clear filament better, but again, this is not the ideal material to use to troubleshoot your machine.
- **Step 7:** Try your print again. Be sure the first layer goes down in a flat, thin bead! If not, you probably have leveling problems (or less commonly, but still possible, heating problems).
  - MakerWare's Medium Default settings (layer thickness of 0.27 mm) are recommended for slicing, to eliminate other variables.
  - If you are having success printing at 0.27 mm, but failures at finer layer thicknesses, —and you have installed an upgraded extruder mechanism—especially experiencing "air printing" further into a print, the following may be occurring:
    - Your upgraded extruder either is not operating smoothly, in which case disassembly, sanding and lubing the hinge is needed. Note that extruder upgrades printed in PLA are prone to warping over time. This is why the ABS parts are better suited for upgraded extruders.
    - The spring may be too strong for the print speeds if you are seeing skipping or hearing 'clicking' from the drive motor.
    - Print speeds may be too slow and heat soak is causing the filament to soften and the drive wheel to slip. Increasing the print speed causes the filament to move more quickly, thus soaking up less heat.
    - For those not using MakerWare, but ReplicatorG or another slicer, some users report improvements by reducing the retraction values.
- **Step 8:** If the print fails when using MakerWare's Medium Default settings, you may have a clogged nozzle, or heat problems. Clogs are evident if the extrusion is not coming down perfectly straight, or is very thin. Measuring the extrusion should indicate a value slightly larger than 0.40 mm.

[23] *http://www.makerbot.com/support/replicator2/documentation/maintenance/* and *http://www.youtube.com/watch?v=QOJN_8AAC9U*

- If significantly less than 0.40, you must clean or replace the nozzle. *[See Clogged Nozzle, below]*
- After cleaning or replacing the nozzle, repeat all steps above.

- **Step 9:** Head temperature issues are the prime remaining suspect if all of the above checks out. These are notoriously difficult to track down for one simple reason—if there is an error, but the Temp Error message is not activated, the Replicator2 overrides the output to the heating core to prevent a fire. However, the display may no longer accurately display actual temperature information.[24]
  - Check the thermocouple wire going into the head. This is in a very vulnerable location and early units had shrink-wrap at the end that was too long, thus this wire was sometimes damaged during assembly. Check this at the hot end. Gently wiggle it to see if there is any play. Ideally, one will get a Temp Error message when this is loose, but not always.
  - If you have received a Heater Temp Error, you must replace the thermocouple wire, being very careful when replacing it.
  - If you have not received a Heater Temp Error, power cycle the machine, leaving it off for one full minute. Then restart and try printing again.
  - If the print fails again, check all wires to the heater core very carefully. Even a small nick in the sheathing may cause electrical fluctuations to trip the over-ride. Call MBI support to discuss additional troubleshooting or repair. Typically, this involves replacing the entire hot end and mounting bar.

2. Shrinkage, Lifting & Curling Problems:

- **Shrinkage of PLA:** For precision work, it is useful to keep in mind that PLA shrinks slightly. Shrinkage is about 0.2% (or one-fifth of one percent). Scaling one's model up before slicing it in MakerWare may be useful if precision fitting is involved.
- **Lifting & Curling:** Tests have shown that lifting or curling of the print (off of the build plate) is a common problem—especially if the print is very long in the X direction. This is due to shrinkage of the model when cooling and an inherently cooler build plate.
  - Even if you have good adhesion at the outer limits of the print (at the far left and far right of the build plate) the acrylic plate will actually deflect and be pulled upwards while printing as the model cools and shrinks.
  - If the Replicator 2 had a heated build plate (similar to the Replicator 2X), this might help counteract or possibly eliminate the problem. Thus, this appears to be an inherent limitation with the Replicator 2. Keeping this challenge in mind may be useful when digitally modeling and/or orienting large prints.
  - Try reducing the number of shells if you have frequent curling.
- **Helper Disks:** One method to help counteract lifting and curling is to use Helper Disks[25] in MakerWare (as of version 2.0.0.161). File » Examples » Helper Disks. Placing these flat disks at the corners increases the surface area of the print on the plate, resisting lifting and curling. The Helper Disks may be peeled away or cut off after the print.

---

[24] *This is an area that should be corrected in the future. If temp over-rides are tripped the LED display should indicate this, instructing a power cycling for further testing and or correction.*

[25] *Helper Disks were originally designed by ctheroux at* ***http://www.thingiverse.com/thing:51066****.*

- **Glass Plate Upgrade:** As mentioned earlier, some users report significant printing improvements after replacing the build plate with glass (done to improve flatness) and using a heavy coat of AquaNet hairspray on the glass. *[See MODS section]*

3. Stripping:

- **Definition:** This occurs when the drive wheel is working properly, but the warm filament is not advancing or retracting. No clicking is heard during this type of failure. Much like a car tire spinning in the mud without the car moving, a rut is created in the filament strand. The more the drive wheel spins without the filament moving, the deeper the rut becomes and the less likely the filament will advance or retract properly.
  - The stock delrin plunger is especially prone to this condition, the thinner the layer being printed, the more likely that filament stripping will occur.
  - This is due to the fact that very small drive wheel movements are being made on the filament while it is absorbing heat from the hot extruder below; thus softening. The drive wheel starts to slip on the softened filament, carving into it deeper and deeper.
  - Symptoms of this are often seen in a printed model which has failed part way through the print. The last few layers are often fuzzy, the extruder continues to move but stops extruding (aka "air printing"), and there is no 'clicking' heard from the drive motor.
- **Identifying stripping:** If your print head shows the proper temperature, but is not extruding when it should be—during a print, or when you try to Load or Unload—you likely have a stripped filament if there is no 'clicking' heard from the head. If there is 'clicking' and the nozzle is not touching anything, your nozzle may be clogged, or you have too much pressure against the filament.

  [If you have the OEM plunger, as noted earlier, your delrin plunger pressure is likely too tight and your extruder is not clogged — especially if the unit is relatively new.]

  - **Step 1:** Run the Unload routine, gently tugging on the filament with the thumb lever fully depressed after the head has heated.
    - If you do not fully depress the thumb lever, you may snap off straggling 'blobs' of molten plastic, causing a jam.
  - **Step 2:** If you are unable to remove the filament, or if it breaks while pulling, proceed to the next section on **Filament Jams.**
  - **Note:** Loading when the filament is stripped is not recommended since it is difficult, if not impossible, to perform successfully with a stripped filament. By attempting to Load, you may create a severe jam by forcing more filament in, breaking the weak spot in the filament and, in essence, pushing new filament on top of the jammed filament beneath the drive wheel. *[See the following section for images of a jam]*

4. Filament Jams:

- **Definition:** A filament jam is caused by a piece of filament breaking off in the space between the drive wheel and the heated chamber, blocking new filament from entering the extruder head.
- **Common Causes of Jams:**
  - Pushing too hard on the filament while Loading.
  - While Unloading, failing to assist the filament. The proper method is to *very slowly* and *gently* pull filament straight up and out, while the thumb lever on your extruder is held down. Be aware that there will generally be a wider blob at the very end. Be careful to not break this off.
  - Pulling on the filament *too hard* while Unloading; thus ripping the filament apart, or leaving "the blob" behind, below the drive wheel.
  - Not fully depressing the thumb lever on the upgrade extruder mechanism while Unloading, thus catching "the blob" and tearing it off.

The best method to prevent jams while Unloading is to pull the white plastic tube from the extrusion assembly and, with your fingers, very gently assist the filament straight out while fully depressing the thumb lever until the filament is removed.

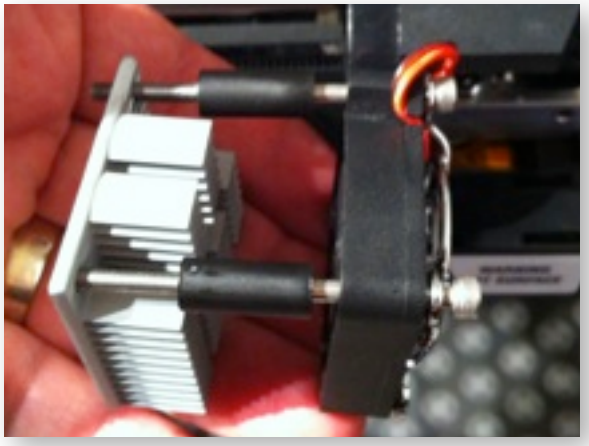
- **Fixing a Jam:** Instructions are below. *[Note that a stripped filament will also need to be corrected the same way as a jam]*
- **Step 1:** On the front cooling fan of the extruder assembly, remove the two allen screws (turning CCW).
  - Be careful withdrawing these long screws since there are two black plastic spacers hidden inside between the fan and the heat sink.
  - When these screws are removed, the drive motor is now loose, as well.

THIS FAN IS BACKWARDS

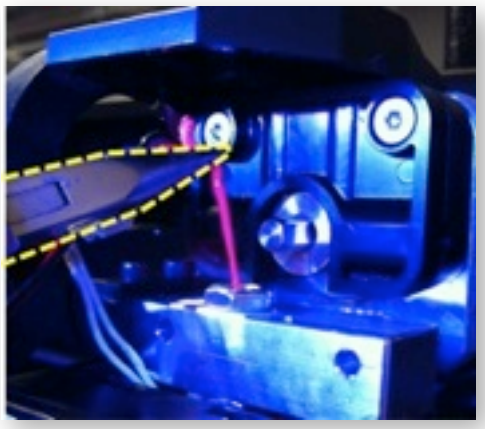
and blows out.

If you can read the label on the fan REVERSE IT TO BLOW IN!

- **Step 2:** In the images below (with the fan shown upside down, so you can see the screws), note that the screws pass through two black plastic spacers, then through an aluminum heat-sink.
  - Cleverly, these screws perform quadruple-duty. What is not visible is that the screws also pass through the aluminum cooling and mounting block, then draw the drive motor assembly tight against the cooling block.
  - Note the position of the wiring as you remove the fan. It should be free of the fan and not be pinched anywhere.



- **Step 3:** The image below (left) shows a serious jam. Less severe jams are common and equally debilitating to the filament loading process.
  - The filament sticking out of the head has been bent by the drive motor.
  - To clear a jam you may trim the filament close to the hot end, or carefully pull it out of the extrusion chamber with a pair of needle nose pliers, and trim it.
  - If you are having difficulty—and as a last resort—with great care you may preheat the extrusion chamber to permit withdrawal of molten filament. Be *very careful* of the HOT extrusion chamber, the fan, and the routing of wires. Gloves are recommended.
  - Be sure to clean the little teeth in the drive wheel!



5. Clogged Nozzle:

- This is a somewhat rare issue (provided you keep a clean work area) and is not addressed in detail here. Unless a particle was embedded in your PLA during manufacture (rare, but possible, especially with very cheap filaments), PLA is burning inside the extruder if you are running temperatures higher than 230°C. Thus, deposits build up which block the hole in the extruder, or reduce the passageway of molten filament. This is not unlike a person's arteries with plaque building up in them, reducing flow and increasing pressure.
  - If you encounter this condition frequently, likely you are running your extruder at too high a temperature. *Temperatures above 230°C are unnecessary and to be avoided.*
  - MBI uses 230°C as the default head temperature; however, PLA will extrude and bond (at non-accelerated speeds) at temperatures as low as 190°C.
  - More experienced users often use temperatures between 210°C and 220° to avoid clogging and to assist in removing small models from the build plate. This is because cooler temps reduce adhesion to the plate slightly.
  - For details on cleaning a nozzle see **Nozzle Cleaning Tool**[26] on Thingiverse, by whpthomas.
  - Worth noting is that the lower the temperature, the less *shiny* the surface of the model will be. This is useful if you are trying to achieve more uniform or matte surface finishes. A temperature of 210°C works well to produce matte finishes for white filament.

6. Squeaking or Screeching:

- This indicates a lack of lubrication, which is normal after ± 50 hours of operation. The X and Y metal guide rails and the Z direction threaded screw should be wiped with silicone grease every fifty hours, or so, of printing.
- A couple drops of oil should be applied to the bearing surface of the black plastic toothed-roller on the left side of X drive belt.
  - *If you accidentally get oil on the drive belt, clean it off immediately with rubbing alcohol.*

26 ***http://www.thingiverse.com/thing:56597***

» INITIAL SETUP: RECOMMENDED BEFORE YOU START PRINTING

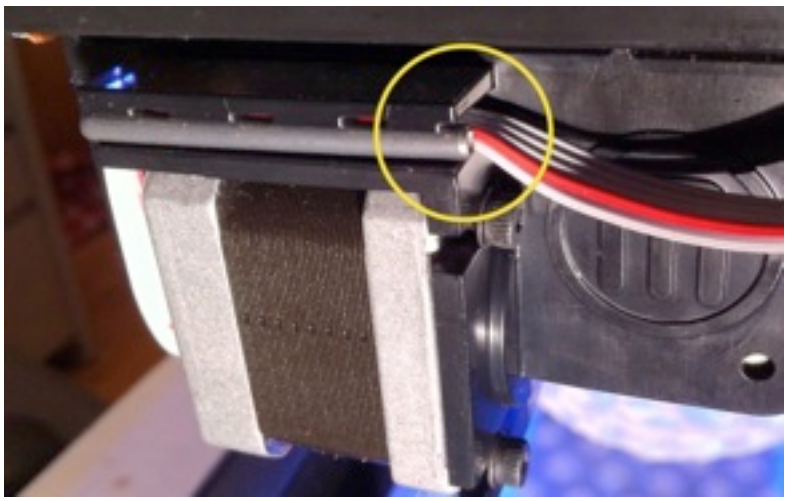
1. Check All Fasteners:

Inexplicably, a number of users have reported loose fasteners on their newly delivered Replicator 2 units. While it is hoped that this simple matter is soon corrected by MBI, at the time of this writing, this problem persists.

- Check and tighten the following:
  - The two bolts on the bottom of the extruder carrier.
  - All bolts which hold the drive motors in place. Be sure that belts are tight.
  - Lower cooling fan should be screwed tight into the black plastic housing. These will be difficult to turn initially because they are self tapping.
  - Any other bolts that relate to moving parts.

2. Remove Spring Pin Holding Cables for X Drive:

A number of users have reported cable chafing and resulting failures for the X axis drive due to contact with the slotted spring pin which holds the cable in the housing.[27] This is a poor use of a spring pin by MBI since these pins are, A) intended for holes, and B) are sharp on the ends to permit entry into said holes. In addition, the slot of the pin—which can also be sharp—usually faces the cables. This is a recipe for disaster, sooner or later, and is best addressed before using the machine.

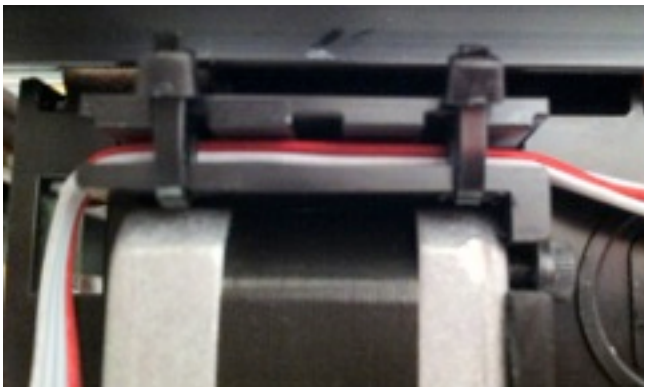


- **Removal of the spring pin is recommended before you begin using your machine to prevent possible damage to the X drive cables**
- You may retain the cables in the slot (to prevent accidental damage) in one of the following ways.

[27] *Thanks goes to Jetguy, on the **MakerBot Google Group** for identifying this problem.*

- Remove the six allen bolts *(5 mm allen wrench)* holding the right side panel.
- Remove the spring pin.
- **Macgyver Method:**
  - Use two small binder clips around the outside of the ABS housing to retain the wires and remove the wire handles on the binder clips.

  --- OR ---
- **Meddler Method:**
  - Remove the four allen bolts *(2.5 mm allen wrench)* holding the X drive motor.
  - Slide the motor out of the slots and move the cables out of the way *(note routing)*.
  - Carefully drill two 3/32" holes up through the ABS carrier *(near the front edge).*
  - Install two small zip ties (3/32" wide x 4" long) to keep the cables in place.
  - Verify the full range of motion (in the Y axis) before you tighten and trim the zip ties—movement should not tension the cables.
  - Reinstall motor, firmly tensioning the belt.



- Reinstall the right side panel.

3. Proper Cooling Fan Orientation:

Several users have reported that their Replicator 2 arrived with the front cooling fan installed so it blew OUT *(away from the heat sink)* rather than blowing IN *(towards the heat sink).*

- If you can read the label on the fan, it is likely blowing out and should be reversed.
- To reverse the fan, simply unbolt the lower screws and flip the fan, being careful to route the wires appropriately.

4. Prepping the Spool Holder & Spools:

Smooth delivery of the filament is crucial for reliable printing. Unfortunately, the design of the spool holder is less than optimal because it has a fairly crisp lip on the top. With the majority of the weight of a spool bearing on this lip, spools need to have ultra-smooth interiors for smooth delivery. **Unfortunately, the spools are not always smooth inside.**

- Using sandpaper or a file, round off the crisp edges on the top of the spool holder.
- Before mounting a new spool, check the interior smoothness, sanding as necessary.

5. Oiling the Filament Feed Tube:

To further assist with smooth delivery of the filament, it is recommended that users place a few drops of vegetable oil on a section of filament and run it back and forth through the filament tube to lubricate it.

6. Proper Drive Wheel Adjustment:

The set screw on the drive wheel is incorrectly installed, potentially causing the drive wheel to slip. The drive motor shaft has a flat ground face which the set screw on the drive wheel is meant to firmly engage. However, this slot is not long enough and the set screw is tightened onto the round part of the drive shaft. Since the shaft is made of hardened steel, the engagement of the screw is poor and prone to slipping, leading to skipping of the filament during the print.

- Disassemble the extruder head *[as described in the Filament Jam section above].*
- Remove electrical wires from the drive motor and remove motor from assembly.
- Loosen the set set screw on the drive wheel and remove the drive wheel from the shaft.
- Using a small flat file, or a dremel, extend the flat slot on the motor shaft an additional ± 3/16" [± 5 mm].
- To reduce heat transfer between the cooling block and the drive motor, a recommendation from whpthomas is recommended—insert two pieces of cardstock (cardstock from a six-pack carrier works well), cutting away the center and the mounting holes.
- Reassemble the drive motor and tighten the set screw on the wheel, being careful that the drive wheel is centered on the bearing of the upgraded extruder, and is directly above the inlet on the hot end. *[An easy method to use is to insert a section of filament to visually aid in centering].*

» MODS: OPTIMIZING AND HOT-RODDING YOUR REPLICATOR 2

In this section, upgrades are described for more reliable and high-precision printing. In particular, the Extruder Upgrade and Glass Build Plate are virtually necessary for fine resolution printing of 0.10 mm, optimal adhesion for large parts, and lengthy prints without failure.

All users are strongly advised to install the Extruder Upgrade described below!!!

1. Extruder Upgrade: **AN ESSENTIAL MOD**

As indicated throughout this tutorial, the stock extruder mechanism (an allen screw controlling a delrin plunger & o-ring) is believed to be inadequate for longer prints at fine resolutions. While you may achieve prints of Low (0.34 mm) and Medium (0.27 mm) thickness, it is unlikely that you will achieve High (0.10 mm) thickness prints for large objects, resulting in "air printing".

DigiFabLab, like many other experienced users, believes that installation of an upgraded extruder is the single best improvement one can make for reliable printing.

- Visit Thingiverse.com, download, print, and install upgraded extruders
- **OPTION 1:** Grab the **Replicator 2 Extruder Upgrade**[28] designed by whpthomas
  - You will need to purchase the hardware specified.
  - If you print the items in PLA, keep an eye on them, since heat warps PLA. You will likely have to replace these parts every now and again over time.
  - Installing two layers of card stock behind the upgraded lever assists with reducing heat transfer and warping (suggested by whpthomas). Cardstock from a six-pack carrier works well. Be sure to move the drive wheel out the same distance as the thickness of the card stock.
- **OPTION 2:** Grab MakerBot's new derivation of whpthomas' design, from Thingiverse.com, or from their store—the **Replicator 2 Extruder Alternative**.[29]
  - Purchase the upgrade hardware from the **MakerBot Store**.[30]
  - Cost for bearing, spring, and screws is $8 USD.
  - Be sure to request the ABS parts, since printing them in PLA on a Rep2 creates parts that are prone to warping from heat.

---

[28] *Replicator 2 Extruder Upgrade by whpthomas (aka Wingcommander) **http://www.thingiverse.com/thing:42250.** This was derived this wonderful and essential upgrade from emmet, infinityplusplus, & whosawhatsis*

[29] *Replicator 2 Extruder Alternative by MakerBot **http://www.thingiverse.com/thing:53125**. Derived from whpthomas, et al.*

[30] *http://store.makerbot.com/parts-accessories/replicator-2-repair-parts-accessories.html*

- On this particular part, it is important that the shoulder bolt seat firmly against the drive block. If not, it will loosen each time you use the lever.

- **OPTION 3:** Just released is an extruder upgrade in aluminum from WotzBotz which (obviously) will not deform due to heat! Cost is ± $99 + shipping. See **here**.[31]
- Be sure that the lever operates smoothly and freely. File or sand parts as needed. Do not over-tighten the hinge mechanism. Silicone grease at the hinge is advised.
- Whichever upgrade you choose, you'll be VERY glad you installed it!

2. Glass Build Plate:

This is the second most important upgrade you can make if you're having adhesion problems and are sure you've set the gap (0.10 mm) correctly. This upgrade is virtually essential if you are interested in large prints (which span the build plate) or have fine layer resolutions (0.10 mm — 0.15 mm). Upgrading the (often) warped MBI acrylic build plate with glass offers significant improvements in adhesion and accuracy. Prepping the surface with a heavy coat of Aqua Net Unscented hairspray seems to work well. Cost is ± $10 to ± $75 USD depending on the thickness of glass desired.

- **Dimensions:** Rectangular glass sheet (which will fit between the tabs) measures 11.25" x 6.375".
- **Macgyver Method:** Have your local glass or hardware store cut a piece of single pane window glass. *[± 2.22 mm thick]* and place this on top of your MBI build plate.
  - Also available from most custom glass shops are one-sided acid-etched sheets. *[± 3.36 mm thick].* The benefit of etched glass, other than increased strength due to greater thickness, is that adhesion is stronger than on smooth sheets. The drawback is that setting one's clearance gap is more difficult due the greater thickness and the resulting spring tension.
    - If installing one of these thicker sheets, placing a small shim on the back of the table, under the Z limit switch is advised to permit proper leveling.
  - Place the glass sheet on top of your MBI acrylic plate.
  - Using a continuous strip of 1" masking tape on the left and right edges, loosely wrap the tape around the sides and to the bottom of the stock acrylic build plate. You don't want to bind it too tightly—you're simply trying to prevent the glass from shifting.
  - If your MBI build plate is very badly warped, tightly binding the glass may cause it to also warp slightly.
- **Meddler Method:** Replace the entire MBI build plate with thick glass. For reference, the stock build plate is about 8.40 mm thick. 3/8" (0.95 mm) thick glass works well.
  - You will have to devise a method to secure the glass in place around the existing tabs on the table unless the glass has been cut to match the MBI profile.
  - If your total assembly (from the plastic of the bed to the top of the glass), exceeds 12 mm, you may not be able to adjust your clearance without altering the Z home limit switch.
  - Check the **MakerBot Google Group** to see if anyone is offering custom shaped plates.

[31] *http://www.wotz101.com/wotzbotz/index.html*

- Or, for the very industrious, make a cad file of an exact copy of the existing plate and have a 3/8" thick piece of glass water-jet cut by a local supplier to fit into the existing tabs.
- Silicon gasket material or pads can be used to keep the glass in place on the table.
- Worth noting is that some have speculated that the glass is best not locked into place due to reasons of thermal expansion. DigiFabLab has not tested this theory, but it's an interesting one.

- **Preparing the Glass Surface:** Clean surface with rubbing alcohol. Spray a heavy film of (Aqua Net unscented) hairspray on smooth glass; a lighter film on etched glass.

3. Sailfish Firmware:

The good folks at Jetty Firmware have created a spectacular firmware upgrade for the Replicator 2. This upgrade permits improved printing speeds, using acceleration, and makes the Rep2 feel like a professional grade printer. The firmware works well with MakerWare and ReplicatorG slicers (and probably any other that generates .X3G files). This upgrade is highly recommended and it's free.

- Among the many excellent features are:
  - temperature and speed over-rides while printing
  - filament used
  - time remaining
  - bed leveling by simply moving the extruder head by hand
  - and far, far more!

4. Aluminum Table Arms:

The ABS plastic table arms are prone to vibration, deflection, and possible sagging. Sagging is perhaps less of a problem with the Rep2 (which is not enclosed), but seemingly likely for the enclosed Replicator 2X (or if one encloses their Replicator 2). Bottleworks is offering **aluminum table arm upgrades**[32] on the MakerBot Operators Group. For users interested in exploring precision printing, this upgrade makes a good deal of sense. Cost is ± $125 USD + shipping.

5. Aluminum Hot End Carriage Mount:

The ABS carriage mount is prone to deformation at higher temperatures. If you plan to print in nylon, this presents a significant challenge. Carl Raffle has created an aluminum carriage mount for the Hot End which is currently being fabricated. See **here**[33] for his post on MakerBot Users Group. Cost is ± $160 USD including global shipping.

[32] *https://groups.google.com/forum/#!msg/makerbot/wGLpDVxcpqM/12u74HDDIBwJ*

[33] *https://groups.google.com/forum/#!searchin/makerbot-users/carriage/makerbot-users/rxthvz49wa0/Sv766tBj_7EJ*

» TOOLS & SUPPLIES, SOFTWARE, AND RESOURCES:

1. Recommended Tools & Supplies:

- Small diagonal cutters (aka "dikes")
- Small needle nose pliers
- Flat and Phillips head screwdrivers
- Razor scraper and razor blades
- Thin putty knife or craft spatula
- Digital calipers (that read mm and inches)
- 7 mm wrench (for nozzle replacement)
- Isopropyl Alcohol 91%
- AquaNet Hairspray (if using a glass build plate)
- Blue painter's tape (3M #2090), possibly worth experimenting with.

2. A Software Sampling:

Software is a highly personal choice, largely dependent upon one's skills and goals. The following is an incomplete sampling of programs commonly used for both Mac OSX and Windows. Software over $1,000 is not listed. Ultimately, the best software is the one that helps you create what you want, in ways you like. It needs to create watertight models and export in .STL format. *Effort = Reward.* Got a program you love? Let us know about it.

- **Netfabb**. *Available in free and pro versions, this does incredible things for repairing models which are not watertight.*
- **Meshmixer.** *A wonderfully fun environment to play with sculptural shapes. Very "kid friendly".*
- **Blender.** *The big-daddy of free open source 3D modeling software. You can create anything you want, but the learning curve can be a bit steep due to differing conventions.*
- **SketchUp.** *Another heavy hitter in the free software area. Can also purchase pro versions. Very simple but somewhat limited capabilities. Watertight modeling can be a bit elusive with complex shapes.*
- **AutoDesk 123D Design.** *Relatively new and free. Can be used online, on an iPad, or on your computer. Getting good reviews.*
- **AutoDesk Fusion 360.** *Cloud based, in beta currently. Based on Inventor. Fees unknown at this time.*
- **ViaCAD 2D/3D.** *$99. Many speak highly of this solid modeler. Pro versions available.*
- **Mol3D.** *$295. Powerful and simple. Works well with pen interfaces.*
- **Bonsai.** *Free for students, $495. A newer arrival from the makers of FormZ, one of the original surface modeling innovators.*

- **Rhinoceros.** *Free trial Windows. Free Mac Beta, $995 Windows. $195 student license. Exceptionally powerful surface modeler with solid features. Highly developed NURBS modeling, but weaker Polygon modeling. Also offers Grasshopper visual scripting plugin. Great features include colored backfaces (in Windows version) and naked edge analysis. Rhinoceros is employee-owned, rock solid, and has a strong user community. This program is DigiFabLab's workhorse software.*

3. A Few Resources:

- ***"Getting Started with MakerBot"***[34] by Pettis, Kaziunas-France, and Shergill. Published by O'Reilly Media and "MAKE" magazine, 2013. *Useful information, tips and projects.*
- **Thingiverse.com**.[35] *Thousands of great projects that you can download and print.*
- ***Make* magazine's *Ultimate Guide to 3D Printing***,[36] December, 2012. ($9.99). *A superb shootout comparing a number of FDM printers, mentions of other style printers, and numerous projects.*
- **RepRap.org**.[37] *This is where it all started and continues to thrive. Be sure to check out the forums.*
- Google Groups: **MakerBot Operators**,[38] (moderated by MakerBot), **MakerBot-Users**,[39] and **3D Printer Tips, Tricks and Reviews**.[40]
- **MakerBot Wikia**.[41] Fledgling wiki created by and for MakerBot enthusiasts.

[34] http://shop.oreilly.com/product/0636920026723.do
[35] http://www.thingiverse.com
[36] http://www.makershed.com/Make_Ultimate_Guide_to_3D_Printing_p/1449357377.htm
[37] http://reprap.org/wiki/Main_Page
[38] https://groups.google.com/forum/%23!forum/makerbot
[39] https://groups.google.com/forum/%23!forum/makerbot-users
[40] https://groups.google.com/forum/%23!forum/3dprintertipstricksreviews
[41] http://makerbot.wikia.com/wiki/Replicator_2

» FAQ's: FROM MAKERBOT'S WEBSITE

*The items below are excerpted from **MakerBot's support website** for convenience.*

1. My build is the wrong size:

- Sometimes the intended size of an object can get lost in translation between your design software and slicing. Our software expects units to be set to millimeters, so make sure that your modeling software is also set for millimeters. If you are not using MakerWare to prepare your objects for building, you may wish to do so, as the Scale button allows you to size models precisely.

2. My Replicator 2 can't see my SD card:

- If your Replicator 2 does not recognize that there is a card in the SD slot, make sure you're using a FAT16 formatted SDSC card no larger than 2GB. If the card is formatted correctly but is unreadable, something may be wrong with the card.

3. I can't find my file on the SD card:

- Your Replicator 2 can't read file names longer than 26 characters. Shorten the file name and try again. Files should end with .X3G

4. Help! My bot sounds like it's shaking itself to pieces!:

- Check to make sure you have acceleration turned on. Attempting to print at high speeds with acceleration turned off can harm your Replicator 2. You can turn it off and on through General Settings in your Replicator 2's onboard menus or through Onboard Preferences in MakerWare or ReplicatorG. If you are running non-accelerated firmware or have acceleration turned off, you should not print at speeds above 40 mm/s. Firmwares from Replicator Firmware 7.0 use a new type of acceleration command, and should only build from .x3g files. Running an .s3g file on a bot running Firmware 7.0 or later is equivalent to temporarily turning off acceleration, and will result in loud, jerky motion.

5. How do I update firmware on my Replicator 2?:

- Select the Upload Firmware option from the Bots menu in MakerWare or the Machines menu in ReplicatorG. The software will walk you through the process. Unlike earlier MakerBots, your Replicator 2 does not have a reset button, so any instructions referencing a reset button can be ignored.

6. Do I need to do any maintenance on my Replicator 2?:

- Not much! But if you've been printing for more than fifty hours or if you start hearing squeaking noises while building then it's probably time to lubricate your threaded rod and idler pulley, and it's also a good idea to clean out your drive gear once in a while. Consult our **Maintenance page** or your **User Manual** for instructions on how to do this.

» ACKNOWLEDGEMENTS, ABOUT DIGIFABLAB, & SHARING:

Acknowledgements:

This guide would not have been possible without assistance from MakerBot Support and the generosity of countless MakerBot enthusiasts found at Google Groups: **MakerBot Operators**,[42] **MakerBot-Users**,[43] and **3D Printer Tips, Tricks and Reviews**.[44] In particular, gratitude is extended *(in alphabetical order)* to: dnewman, Eighty, Enginewiz, Jetguy, and whpthomas/Wingcommander for sharing their efforts, knowledge, and enthusiasm. Without the collective wisdom of all the enthusiasts in this group, the tremendous potentials of the Replicator 2 would be elusive to many.

DigiFabLab is particularly indebted to Norwich University, School of Architecture + Art, which generously loaned one of their MakerBot Replicator 2 printers. This served as a test-bed for the exploration of the Replicator 2's capabilities and for the development of this guide. Faculty and students at Norwich also provided invaluable feedback during three days of in-depth workshops held there during Spring 2013.

About DigiFabLab:

info@DigiFabLab.com

DigiFabLab is a design consultancy dedicated to assisting people with digital design and fabrication topics. David Celento, Founder, is a registered architect with over two decades of university teaching experience. His research and creative works have been presented widely at conferences with numerous workshops conducted globally. David currently serves on the Board of ACADIA (Association for Computer Aided Design in Architecture) and is presently writing a book intended to introduce and demystify digital design and fabrication methods. *Digital Toolbox* (working title) is scheduled for publication by John Wiley and Sons in the Fall of 2014.

Use of this Material:

The information provided in this guide was gathered from many sources and is meant to be shared without charge or restriction. Content may be used in accordance with Creative Commons Licensing version 3.0. Thus, any aspect of this guide may be shared freely; attribution of images and content is requested; and derivative works are permitted—provided they are not for commercial purposes. It takes a village to raise a 3D printer!

---

[42] *https://groups.google.com/forum/#!forum/makerbot*

[43] *https://groups.google.com/forum/#!forum/makerbot-users*

[44] *https://groups.google.com/forum/#!forum/3dprintertipstricksreviews*